

Numerical Simulation and Experimental Study of the Rotor–Stator Interaction of a Turbine Under Variable Flow Coefficients

Ran Ren¹, Qiang Du¹, Guang Liu¹, Zengyan Lian¹, Lei Xie¹ and Yifu Luo¹

Received: 06 September 2023 / Accepted: 17 January 2024
© Harbin Engineering University and Springer-Verlag GmbH Germany, part of Springer Nature 2025

Abstract

Clarifying the gas ingestion mechanism in the turbine disc cavity of marine gas turbines is crucial for ensuring the normal operation of turbines. However, the ingestion is influenced by factors such as the rotational pumping effect, mainstream pressure asymmetry, rotor–stator interaction, and unsteady flow structures, complicating the flow. To investigate the impact of rotor–stator interaction on ingestion, this paper decouples the model to include only the mainstream. This research employs experiments and numerical simulations to examine the effects of varying the flow coefficient through changes in rotational speed and mainstream flow rate. The main objective is to understand the influence of different rotor–stator interactions on the mainstream pressure field, accompanied by mechanistic explanations. The findings reveal inconsistent effects of the two methods for changing the flow coefficient on the mainstream pressure field. Particularly, the pressure distribution on the vane side primarily depends on the mainstream flow rate, while the pressure on the blade side is influenced by the mainstream flow rate and the attack angle represented by the flow coefficient. A larger angle of attack angle can increase pressure on the blade side, even surpassing the pressure on the vane side. Assessing the degree of mainstream pressure unevenness solely based on the pressure difference on the vane side is insufficient. This research provides a basis for subsequent studies on the influence of coupled real turbine rotor–stator interaction on gas ingestion.

Keywords Rotor–stator interaction; Pressure field; Flow coefficients; Unsteady Reynolds-averaged Navier-Stokes modeling (URANS); Attack angle

1 Introduction

The turbine rotor disk cavity in marine gas turbines faces the challenge of high-temperature gas ingestion. To prevent the detrimental effects of gas ingestion on the lifespan of the turbine rotor disk, a sealing flow is introduced to seal the

turbine disk cavity. To ensure an appropriate sealing flow rate, the mechanism of gas ingestion must be understood.

Previous studies have identified three mechanisms that induce gas ingestion in the turbine disk cavity: rotational induction (Owen, 2011a; Sangan et al., 2013b), external pressure induction (Owen, 2011b; Sangan et al., 2013a), and unsteady effects (Beard et al., 2017; Gao et al., 2020). Subsequent research revealed that the circumferential pressure asymmetries caused by the turbine vanes and blades in the mainstream play a dominant role in gas ingestion. A series of studies has been conducted to investigate this phenomenon further. Green and Turn (1994) conducted the first experimental study to examine the combined effect of vanes and blades on ingress, finding that the presence of blades decreases the ingestion. Hills et al. (2002) used unsteady CFD with the model, including vanes and rotating pegs instead of blades, and found that lower circumferential pressure from the pegs than from the vanes can cause more substantial ingestion. Bohn et al. (2003) performed laser Doppler velocimetry (LDV) experiments and found that vanes and blades strongly influenced ingestion. Roy et al. (2001; 2005) measured mainstream time-averaged and transient pressure distributions and found that the blades had the same influence as the vanes on the mainstream asym-

Article Highlights

- To ensure the proper operation of marine gas turbines, elucidating the mechanism of gas ingestion into the turbine is crucial.
- To investigate the impact of rotor-stator interaction on ingestion, a simplified model with only the mainstream passage is chosen to analyze the mainstream pressure distribution under different flow coefficients.
- Using a combined approach of experimental and numerical simulation methods, the pressure distribution patterns under various flow coefficients obtained by altering the mainstream flow rate and rotational speed are mechanistically explained.
- These efforts lay the groundwork for studying gas ingestion in complex turbine disc cavities by establishing boundary conditions.

✉ Qiang Du
duqiang@iet.cn

¹ Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100190, China

metrical pressure distribution. Furthermore, researchers have investigated different mainstream flow coefficients. Bru Revert et al. (2021) investigated the influence of a range of flow coefficients on ingestion by keeping the mainstream flow constant and varying the rotational speed, finding that the sealing effectiveness inside the wheel-space was reduced for increased rotor speeds at constant purge flow rates under axisymmetric conditions, while the inverse phenomenon was observed when the stator vanes were introduced. Palermo et al. (2019) conducted a CFD study with different flow coefficients by fixing the rotational speed and varying the mainstream flow, finding that the vane improved sealing effectiveness compared to the vane-free case at the same mainstream flow. However, the sealing effectiveness of the vane case at high mainstream flow is higher than the vane-free case at low mainstream flow. Scobie et al. (2014) conducted experimental research over a wide range of flow coefficients ranging from 0 to 0.9 and found that the circumferential pressure difference is directly proportional to the flow coefficient, but at lower flow coefficients, severe gas ingestion occurs because of the “blade effect.” The study of gas ingestion with mainstream vanes couples the joint influence of rotationally induced, external pressure induced, and rotor–stator interaction effects under complex flow but has not yet reached a unified conclusion. To simplify the problem, it is necessary to decouple the influence of the disk cavity and clarify the pressure distribution and unsteady flow mechanism of the main flow.

However, previous studies on the mainstream rotor–stator interaction have focused more on cascades and flow structure. Mahallati and Sjolander (2013) found limited inhibition of blade flow separation by wake trails in their study on the cascade. Förster et al. (2012) found strong non-stationarity in the passage vortices in the cascade. Matsunuma (2006) analyzed the details of the flow in the channel at a low Reynolds number and found that the wake was bent and deformed at the blade inlet. Rose et al. (2013) found a considerable nonstationary flow in the inlet and outlet of the blade in the Class 1.5 low-speed axial turbine due to the wake interference. MacIsaac (2010) measured the time-averaged and turbulent flow fields downstream of low-speed turbine cascades and explained the secondary flow loss formation mechanism. Miller et al. (2003) found a complex three-dimensional interaction between the high-pressure stage and the downstream guide vane, in which the guide vane wake and hub passage vortex interactions dominate. Eymann et al. (2002) and Gier et al. (2002) used experiments and numerical simulations to investigate the unsteady effect on low-pressure turbines and to analyze the beneficial effect of secondary flow on losses. Wang et al. (2023) studied the unsteady flow characteristics of the tip leakage flow under the stator–rotor interaction and analyzed the influence of this interaction on pressure oscilla-

tions, mixing, and vortex evolution within the tip leakage flow. Touil and Ghenaïet (2021) investigated the unsteady flow through a two-stage high-pressure axial turbine with analyses of the rotor–stator interaction effects on the aerothermodynamic performance. Zheng et al. (2023) investigated the forced response of an embedded compressor rotor induced by upstream and downstream stator disturbances and rotor–stator interactions through full-annulus unsteady calculations with decoupled and coupled configurations. Duda et al. (2021) investigated the interaction of the rotor blade wakes with the stator ones by changing the stator wheel’s angle using the particle image velocimetry (PIV) technique. Gaetani (2018) analyzed the flow physics for high- and low-pressure turbines and commented on the effects of three-dimensional design on the interaction. Hu et al. (2023) and Yang et al. (2022) investigated the pressure fluctuations of the rotor–stator interaction in a pump–turbine.

Previous studies have extensively investigated the gas ingestion induced by mainstream pressure variations. However, some conclusions remain unclear, such as the inconsistent impact of the blade on gas ingestion and the non-monotonic effect of the flow coefficient on pressure distribution and gas ingestion. To further clarify these conclusions, clarifying the pressure distribution and unsteady flow characteristics initially under only the mainstream rotor–stator interaction is necessary, but most previous studies on the mainstream rotor–stator interaction have ignored the pressure distribution between vanes and blades. Hence, rotor–stator interaction must be studied at different flow coefficients in the turbine stage as a basis for the influence of subsequent rotor–stator interaction on ingestion.

2 Experimental and numerical methods

2.1 Experimental method

2.1.1 Experimental facility

The experimental rig used in this study is an autonomously designed single-stage turbine rotor disk cavity experimental rig at the Institute of Engineering Thermophysics, Chinese Academy of Sciences. An overall schematic of the experimental rig is shown in Figure 1.

The experimental rig is driven by an ABB QABP160M2B electric motor manufactured by ABB company, with a maximum rotational speed of 4 700 r/min and a rated power of 15 kW. The mainstream annulus is supplied by an MJL300 Roots blower, capable of delivering a maximum flow rate of 97 m³/min. In contrast, an MJA100 Roots blower provides the secondary airflow, whose maximum flow rate is 13 m³/min. Because this experiment focuses on the pressure distribution of the mainstream under different operating conditions, the secondary airflow is sealed to prevent

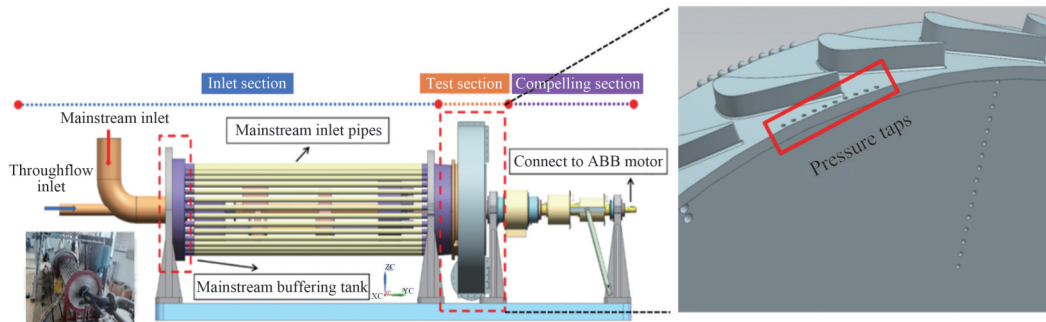


Figure 1 Configuration of the rotating disk cavity experimental rig

any interference from sealing flow and disk pump effects.

The test section comprises a stationary disk with 33 guide vanes and a rotating disk with 66 blades. The diameter of the disk is 500 mm, and the blade passage height is 10 mm. The test section allows for measuring the pressure at the trailing edge of the vanes in the mainstream, as well as the pressure inside the rotor disk cavity and the sealing effectiveness. This experiment aims to focus simply on the pressure field distribution of the mainstream under different operating conditions and to serve as a study of the external boundary conditions for gas ingestion. Therefore, the measurement is solely focused on the pressure at the trailing edge of the vanes.

2.1.2 Operating conditions

Studying the flow mechanisms in the mainstream under various rotational speeds and inlet flow conditions is crucial for gas turbine engines, as they often operate under non-design conditions during start-up, maximum power, acceleration, and deceleration phases. This research aims to provide insights into the flow mechanisms at different mainstream operating conditions, which will serve as vital boundary conditions for investigating gas ingestion. The annulus flow in a turbine is commonly characterized by the flow coefficient C_F . Equation (1) defines C_F . Figure 2 illustrates a schematic of the velocity triangles between turbine stages. C , V , Ωb , and W represent the vane exit velocity, relative velocity at the blade inlet, rotational speed of the rotor, and axial velocity. α and β represent the flow angles at the vane exit and the blade inlet, respectively. According to the design condition of the vane exit airflow angle and the blade inlet airflow angle given in Table 3, the C_F value for the design condition can be obtained from the velocity triangle as 0.64.

$$C_F = \frac{W}{\Omega b} \quad (1)$$

According to the velocity triangles between turbine stages and the definition of the flow coefficient C_F , different values of C_F result in variations in the velocity triangles between stages. Altering the mainstream flow and rotational speed impacts C_F . To investigate the influence of different C_F

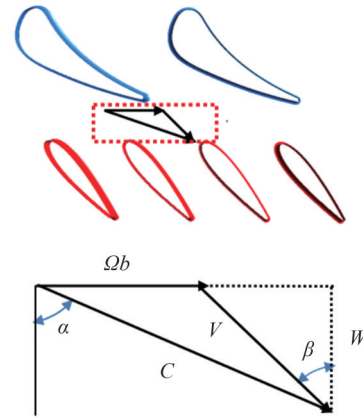


Figure 2 Turbine interstage velocity triangle

values obtained by independently changing the mainstream flow and rotational speed of the mainstream pressure field, the following experiments were conducted: (1) experiments under variable operating conditions with a fixed rotational speed of 4 000 r/min and varying the mainstream flow rate; and (2) experiments under variable operating conditions with fixed mainstream flow rates of 1.272 5 kg/s and 0.994 12 kg/s and varying the rotational speed. The specific operating conditions corresponding to each experiment are outlined in Tables 1 and 2.

Table 1 Operating conditions at a fixed speed at 4 000 r/min

Rotational Reynolds number, Re_ϕ	1.69×10^6	
Axial Reynolds number, Re_w	$(0.507-1.3) \times 10^6$	
Flow coefficient, C_F	0.3–0.77	

Table 2 Operating conditions at a fixed mainstream flow

Mass flow (kg/s)	1.272 5	0.994 12
Axial Reynolds number, Re_w	1.08×10^6	8.45×10^5
Rotational Reynolds number, Re_ϕ	$(0.847-1.69) \times 10^6$	$(0.847-1.69) \times 10^6$
Flow coefficient, C_F	(0.64–1.28)	(0.5–1)

2.2 Numerical method

2.2.1 Computational models

To investigate the unsteady flow mechanisms related to rotor–stator interaction, the numerical study chooses the single-stage turbine without the disk cavity based on the dimensions of the experimental rig. Figure 3 shows the computational domain model of the mainstream passage for the single-stage turbine. The vane profile is derived from the root section of a self-designed engine guide vane model developed at the Institute of Engineering Thermophysics, Chinese Academy of Sciences. The blade profile is NACA0018, a symmetric airfoil with zero lift. Table 3 provides the relevant parameters for the blade profiles.

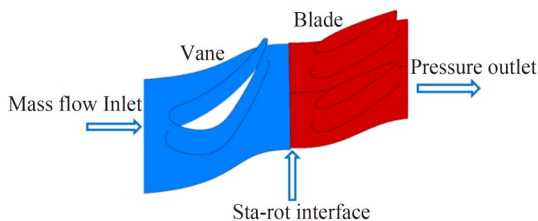


Figure 3 Single-stage turbine mainstream computational domain model

Table 3 Vane and blade parameters

Parameters	Value
Number of vanes/blades	33/66
Vane inlet mach number	0.2
Vane exit mach number	0.65
Vane inlet airflow angle (°)	0
Vane exit airflow angle (°)	67.04
Blade inlet airflow angle (°)	38.41
Blade exit airflow angle (°)	44.71
Vane/blade chord (mm)	46.5/33.10
Distance between vane and blade (mm)	15

To balance computational accuracy and resource requirements, the 10.909° sector model (1/33 of the annulus) of the mainstream passage is chosen for the computation. The mesh for the mainstream passage is generated using ANSYS TurboGrid 19.0. The first layer thickness of the mesh is set to 0.005 mm with a stretching ratio of 1.3. Figure 4 shows the mesh configuration for the vane and blade passages.

2.2.2 Computational setups and conditions

In this study, ANSYS CFX 19 commercial software is employed for simulation purposes. The shear stress transport turbulence model is used. The URANS (unsteady Reynolds-averaged Navier–Stokes) numerical simulation method is

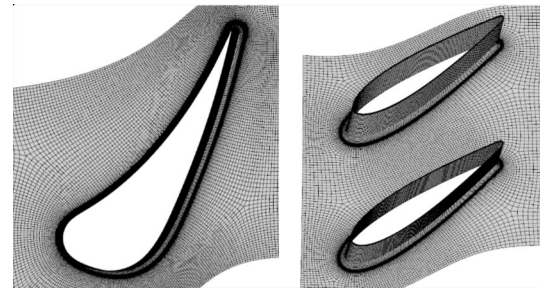


Figure 4 Mesh configuration for the vane and blade passages

employed, with the RANS (Reynolds-averaged Navier–Stokes) results serving as the initial field. The boundaries are defined as the mass flow rate inlet and the pressure outlet. The interface between the guide vane and blade regions is treated differently by RANS and URANS calculations, employing the mix-plane and transient rotor–stator methods, respectively.

According to the velocity triangles, C_F is determined to be 0.64 at the design condition. According to the definition of C_F , the rotational speed and the mainstream flow rate can affect its value. The effects of the mainstream flow rate and the rotational speed on the mainstream pressure were investigated by performing the following numerical simulations: (1) Four numerical simulations were performed by keeping the rotational speed fixed at 4 000 r/min and varying the mainstream flow rate. The flow coefficients considered were $C_F = 0.3, 0.5, 0.64$, and 0.83 . (2) Four numerical simulations were conducted by keeping the mainstream flow rate constant at 1.272 5 kg/s and varying the rotational speed. The flow coefficients considered were $C_F = 0.43, 0.5, 0.64$, and 0.85 . Tables 4 and 5 show the numerical boundary conditions.

Table 4 Numerical conditions at a fixed speed at 4 000 r/min

Rotational Reynolds number, Re_ϕ	1.69×10^6
Axial Reynolds number, Re_w	$0.507 \times 10^6, 0.845 \times 10^6, 1.08 \times 10^6, 1.4 \times 10^6$
Flow coefficient, C_F	0.3, 0.5, 0.64, 0.83

Table 5 Numerical conditions at a fixed mainstream flow

Mass flow (kg/s)	1.272 5
Axial Reynolds number, Re_w	1.08×10^6
Rotational Reynolds number, Re_ϕ	$3.61 \times 10^6, 2.12 \times 10^6, 1.69 \times 10^6, 1.27 \times 10^6$
Flow coefficient, C_F	0.43, 0.5, 0.64, 0.83

2.2.3 Grid and timestep independence validation

To ensure that the calculation results are grid-independent, the $C_F = 0.64$ working condition is chosen, and the grid-independence of 1, 2, 3, 4 million cells is verified. Figure 5 presents the circumferential distribution of dimensionless pressure at the 1.5 mm positions of the vane trailing edge

and the blade leading edge for different grids. Dimensionless pressure C_p (note: when studying the flow coefficients of a fixed mainstream, the mainstream dynamic pressure head is dimensionless) is defined using Equation (2). The figure shows that when the number of grid cells reaches 3 million, further increasing the number of cells minimally affects the peak pressure values at the vane trailing edge and the blade leading edge. Therefore, a grid comprising 3 million cells was selected for the numerical simulations in this study.

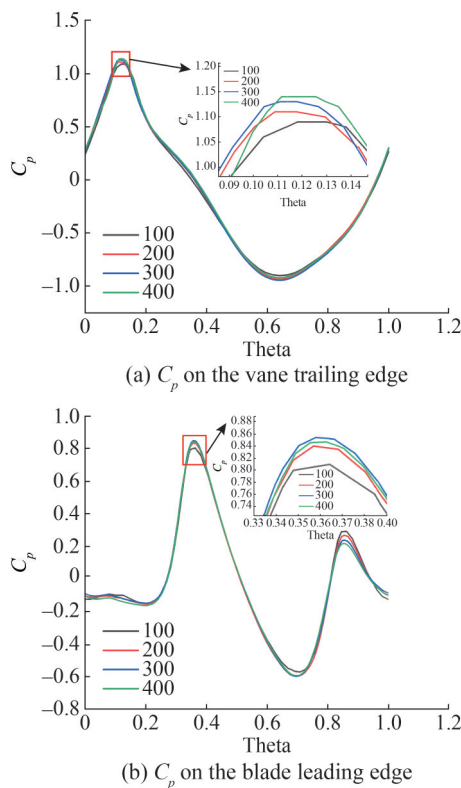


Figure 5 Dimensionless pressure circumferential distribution at the vane trailing edge and blade leading edge

$$C_p = (p - p_{\text{avg}}) / 0.5 \rho \Omega^2 b^2 \quad (2)$$

Considering the inherent instability of the rotor–stator interaction flow in the annulus, a sufficiently small timestep must be selected to resolve all relevant time scales. This decision can be made using the physical CFL. Two different timesteps were tested, 40 steps and 20 steps, to cover one blade passage revolution. Correspondingly, the timestep sizes were set to 5.682×10^{-6} and 1.136×10^{-5} , respectively. The domain-averaged CFL numbers for these two cases are 1.75 and 3.49, respectively. Both CFL numbers are relatively small and close to 1. Considering the computational time and accuracy, the timestep size of 1.136×10^{-5} is chosen for the numerical simulations.

2.2.4 Experimental validation

The experiments conducted on the single-stage turbine

rotor disk cavity rig, varying the flow coefficient by two methods, are used to validate the numerical approach. Figure 6 presents the distribution of measurement points at the location 1 mm downstream of the vane trailing edge in the experimental setup. Figure 7 compares the experimental and URANS time-averaged results at the location 1 mm downstream of the vane trailing edge for two different mainstream flow rates with a fixed rotational speed of 4 000 r/min. Figure 8 compares the experimental values of pressure distribution and the URANS time-averaged results for two speed conditions with a fixed main flow rate of 1.27 kg/s. For the two types of variable conditions varying C_F , the simulation and experimental results are in good overall agreement, and the difference between the peak and valley values of interest and the experimental values is small, verifying the accuracy of the numerical simulation.

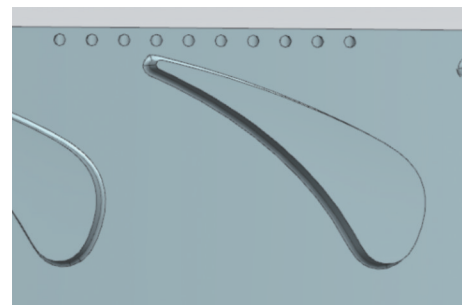


Figure 6 Distribution of pressure measurement points at the 1 mm position of the vane trailing edge

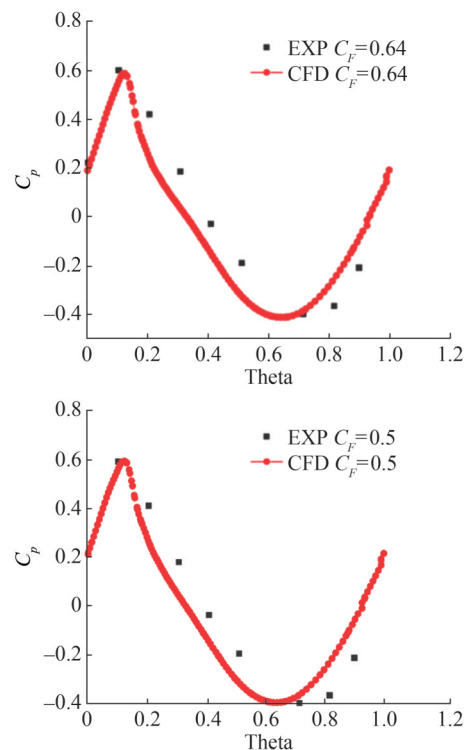


Figure 7 Dimensionless pressure distribution in the annulus compared to experimental data at a fixed rotational speed

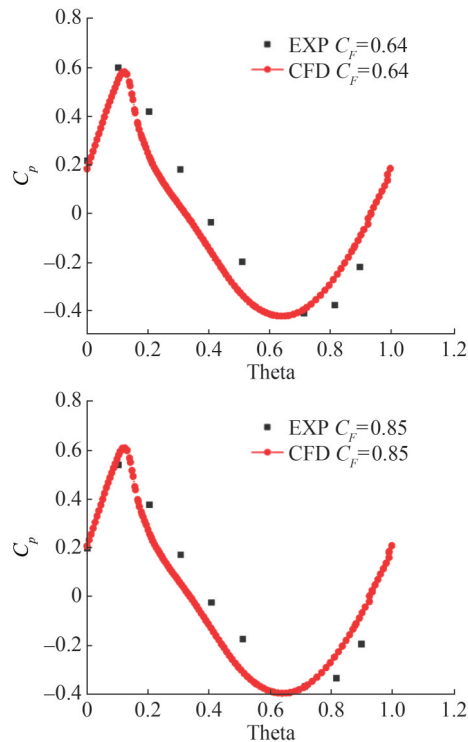


Figure 8 Dimensionless pressure distribution in the annulus compared to experimental data at a fixed mass flow

3 Results and discussion

3.1 Effect of rotor–stator interaction on mainstream pressure

The mechanism of unsteady flow and its influence on the annulus pressure distribution due to rotor–stator interaction is investigated by analyzing a specific operating condition. The chosen operating condition for analysis is a mainstream mass flow rate of 1.272 5 kg/s, a rotational speed of 3 000 r/min, and a flow coefficient of 0.85.

3.1.1 Mainstream pressure distribution

The circumferential variation in annulus pressure is considered a key factor determining the ingestion. Therefore, the annulus circumferential pressure difference must be focused on for the subsequent gas ingestion study. Figure 9 shows the axial distribution of the dimensionless circumferential pressure difference from the vane trailing edge downstream 1 mm to the blade leading edge upstream 1 mm at the 5% span for RANS, time-averaged URANS, and different instants URANS. T represents the blade passing period. The dimensionless circumferential pressure difference is denoted by ΔC_p and defined by Equation 3.

$$\Delta C_p = C_{p\max} - C_{p\min} \quad (3)$$

On the side of the vane trailing edge, ΔC_p obtained by

the RANS calculation is lower than the time-averaged and instantaneous URANS results because the RANS method cannot capture the trailing edge vortex and flow on the vane suction surface well. On the side of the blade leading edge, RANS and time-averaged URANS are similar and lower than the instantaneous URANS results. These values are lower because the RANS and time-averaged URANS methods mitigate the influence of the wake at the interface, which is weaker than the mainstream flow. As the flow passes downstream, it is averaged with the mainstream, leading to unsubstantial differences in stagnation pressure ahead of different blades. For the instantaneous URANS results, the influence of the mainstream and vane wake on the blade varies as the blade rotates relative to the vane. This behavior leads to considerable time variation in the pressure distribution on the blade side. For the chosen condition, the blade potential flow pressure field slightly influences the vane, resulting in little difference in pressure on the vane side.

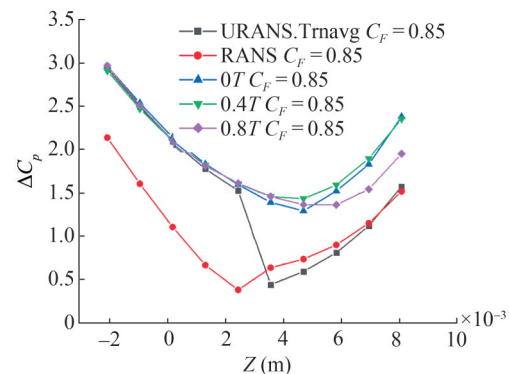


Figure 9 Differential pressure distribution of different simulation methods from the vane trailing edge downstream 1 mm to the blade leading edge upstream 1 mm

Figure 10 shows the pressure and velocity distribution contours at the 5% span. From the pressure contour, the time-averaged URANS and RANS results in the blade of the pressure distribution are identical without the vane wake, as shown by the dotted circle in the figure. For the 0T moment, the blade affected by different wakes and mainstream will stagnate to form different pressures; as in the figure, the blade pressure is higher at the circle position than at the rectangle position. The velocity contour shows the effect of different vane wakes. The blades of time-averaged URANS and RANS results are in the uniform velocity field without a vane wake. At 0T, the blades are located in the wake and mainstream velocity fields with different velocities, as shown in the figure, and the pressure is higher at the blade leading edge located in the mainstream velocity field than at the blade leading edge in the wake velocity field after the velocity–pressure transformation. Figure 11 gives the C_p distribution at the blade leading edge upstream 1 mm for the RANS, time-averaged URANS, and 0T

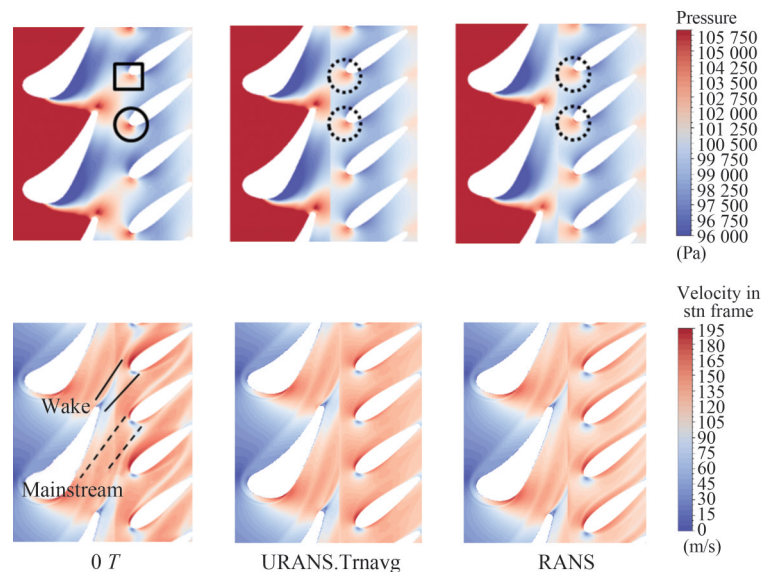


Figure 10 Pressure and velocity contours at the 5% span of the annulus

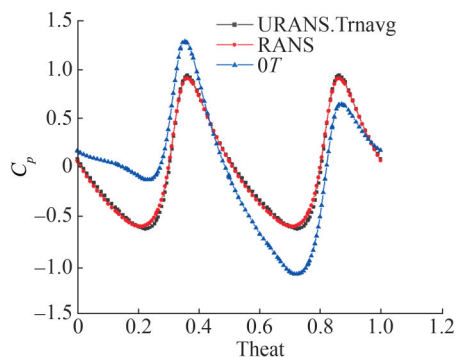


Figure 11 C_p circumferential distribution at the 1 mm position of the blade leading edge

URANS results, which shows the difference in pressure distribution at the blade leading edge due to the presence of the wake. The RANS calculation method does not apply to studying the mainstream rotor–stator interaction, and the URANS method is needed.

3.1.2 Pressure distribution in one blade passing period

To investigate the effects of rotor–stator interaction on the pressure distribution along the end walls over time, the pressure contours at $0T$, $0.2T$, $0.4T$, $0.6T$, and $0.8T$ in one blade passing period are plotted. Figure 12 shows the pressure, velocity, and radial vortex contours at a 5% span position at different times in one blade passing period, where the velocity and radial vortex contours reflect the trajectory of the vane wake. The blade, indicated by the black circle, experiences a diminishing influence from the vane wake and an increasing influence from the mainstream flow during the time interval from $0T$ to $0.4T$. Consequently, the stagnation pressure at the blade's leading edge gradually increases. From $0.4T$ to $0.8T$, the rotation of

the blade gradually decreases the influence of the mainstream flow on the blade and increases the influence of the downstream vane wake. Thus, the stagnation pressure at the blade's leading edge gradually decreases.

To more clearly represent the influence of the rotor–stator interaction on the vane trailing edge and the blade leading edge, Figure 13 shows the dimensionless circumferential pressure distributions at the vane trailing edge downstream 1 mm and the blade leading edge upstream 1 mm. The pressure potential field of the blade at different times has a relatively minor impact on the region near the vane trailing edge. However, the effect is slightly greater on the low-pressure region than on the high-pressure region near the vane trailing edge, as shown in the red circle. The wakes from the vane and the mainstream flow considerably impact the blade pressure field at different times. The pressure near the blade's leading edge exhibits distinct variations over time, influenced by the blade's position within the mainstream flow. When the blade aligns with the mainstream flow, the stagnation effect increases the pressure on the blade side ($0T$ – $0.4T$). As the blade moves from the mainstream and encounters the wake, the stagnation pressure on the blade's leading edge gradually decreases ($0.4T$ – $0.8T$).

Figure 14 shows the distribution of the inlet circumferential average relative airflow angle along the blade height direction at the blade inlet. The blade is influenced by different mainstream and vane wakes at different times, resulting in different pressure distributions, which corresponds to different inlet airflow angles at the blade inlet (attack angles). Figure 13 shows that the different times correspond to different inlet airflow angles, and the most obvious difference is at the 0.15–0.8 blade height position.

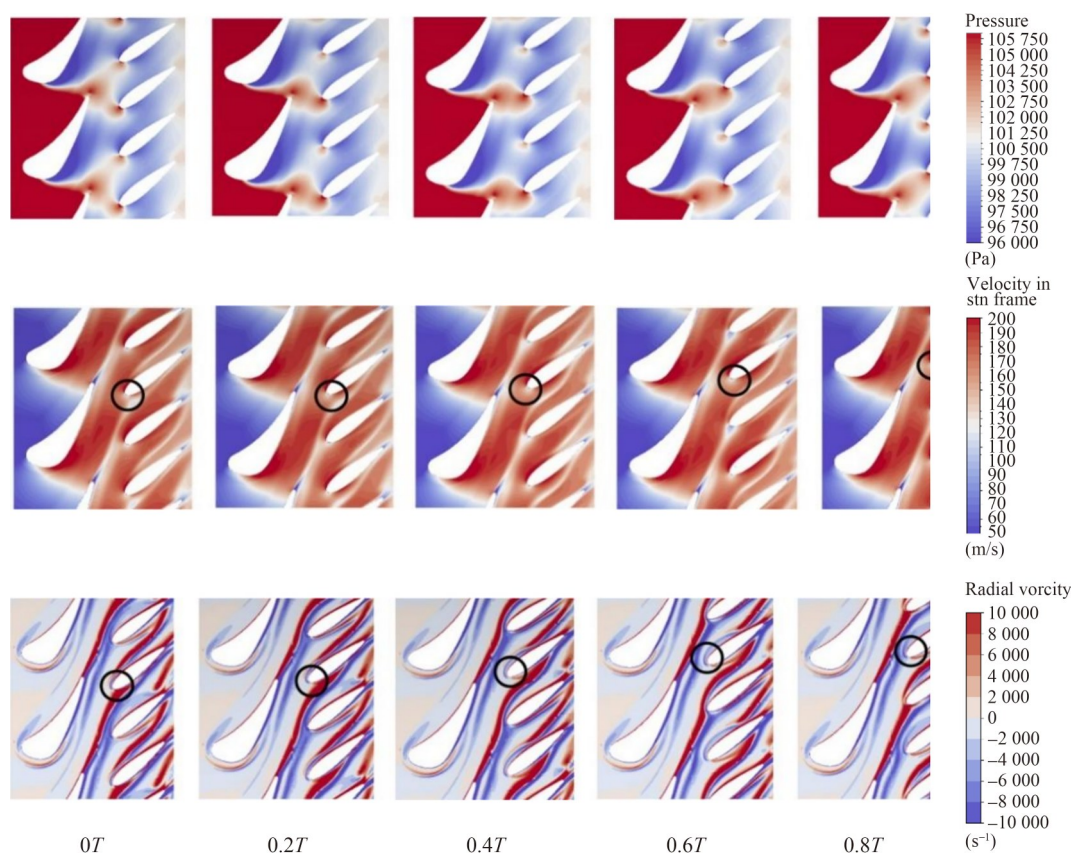
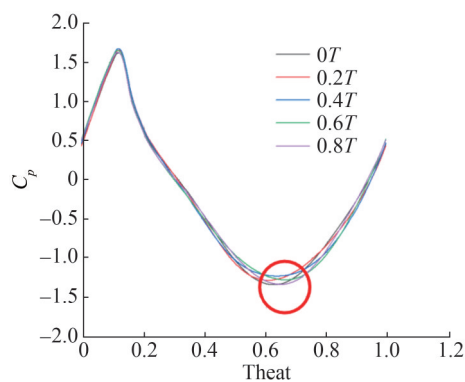
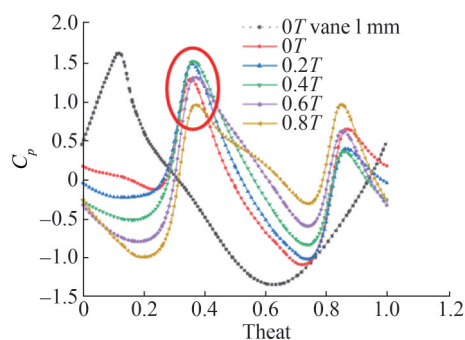


Figure 12 Pressure, velocity, and radial vortex contours at the 5% span of the annulus



(a) Vane trailing edge downstream 1 mm



(b) Blade leading edge upstream 1 mm

Figure 13 C_p circumferential distribution of the 1 mm position at the vane trailing edge and blade leading edge

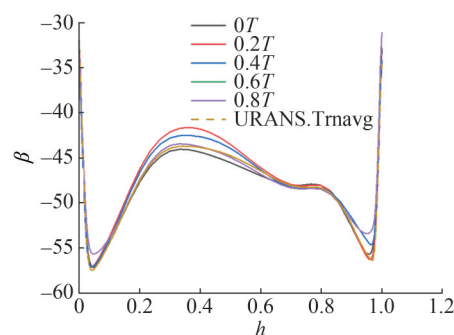


Figure 14 Distribution of the mean relative airflow angle of the blade inlet at different times along the blade height

3.1.3 Mainstream flow field

Figure 15 shows the evolution of the flow field within the vane passage. The oncoming mainstream boundary layer is arrested at the vane leading edge, creating a radial pressure gradient. Under the influence of this radial pressure gradient, the fluid near the end wall rolls up to form a horseshoe vortex. Subsequently, under the pressure gradient along the end wall, horseshoe vortices with opposite directions are formed on the suction side (V_{sh}) and pressure side (V_{ph}) of the vane. Because of the transverse pressure gradient within the passage, the V_{ph} migrates toward the vane suction side, intersecting with the V_{sh} to form a shroud passage

vortex. Unlike other high aspect ratio blade profiles, end-wall passage vortices and a wide range of wake vortices are present at the vane exit. Because this particular blade has an extremely low aspect ratio, the upper and lower end-wall passage vortices fill the entire blade passage and mix with the shed vortices to form a trailing shedding vortex.

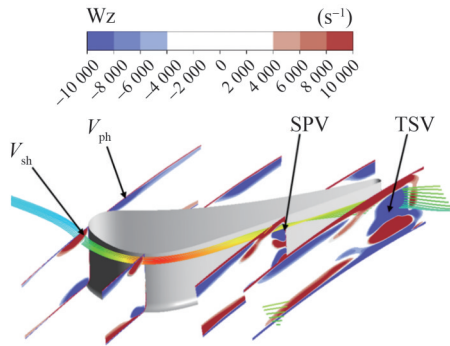


Figure 15 Flow field development in the vane passage

The flow structure in the vane passage affects the blade side. Different blades located at different circumferential positions downstream of the vane experience varying influences from the upstream flow field. This variation results in different pressures at their leading edges. Figure 16 shows the flow field distribution at the blade side at time $0T$ and the corresponding pressure contour. The pressure is obviously smaller for the blade leading edge, mainly influenced by the vane wake, than for that mainly influenced by the mainstream. The pressure at the blade leading edge, influenced by the vane wake, is lower than the pressure influenced by the mainstream flow.

3.2 Influence of C_F (changing the speed) on mainstream pressure

On the basis of the analysis of the rotor–stator interaction mechanism under specific operating conditions, the pressure field caused by rotor–stator interaction under different flow coefficients is investigated. Initially, the pressure field is analyzed for different flow coefficients obtained by changing the rotational speed at a fixed mainstream flow rate.

Although the pressure differs between the time-averaged and instantaneous URANS results under a given operating condition, the time-averaged URANS results can still reflect the trend of pressure changes under different operating conditions. Figure 17 presents the time-averaged URANS results of the axial distribution of the dimensionless circumferential pressure difference ΔC_p between the vane trailing edge and the blade leading edge at the 5% span for different flow coefficients. It is observed that when the mainstream flow rate is kept constant, the pressure on the vane side remains relatively unchanged between rotational speeds. However, the pressure on the blade side initially

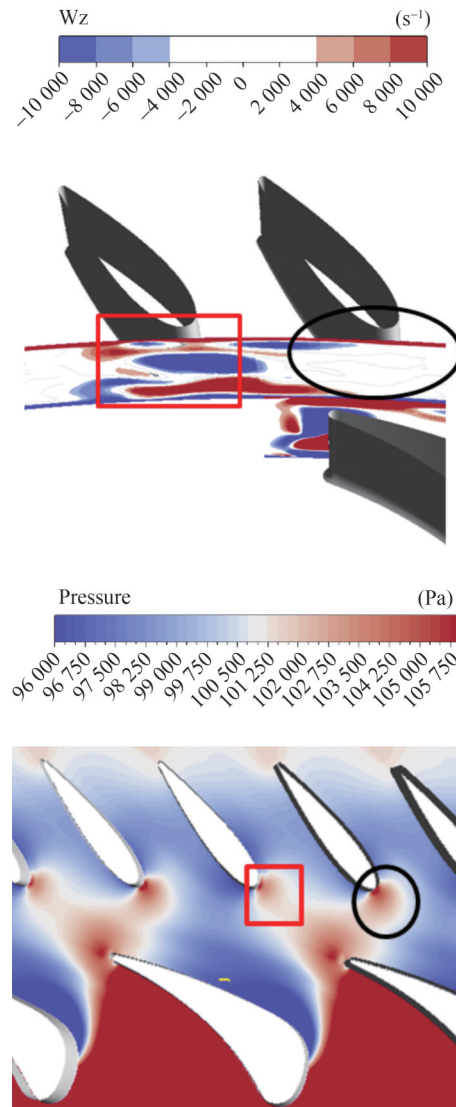


Figure 16 Flow field and pressure contours at the blade leading edge at $0T$

decreases and then increases as the rotational speed increases. The change in C_F changes the airflow angle at the blade inlet, which changes the pressure at the blade leading edge. Figure 18 shows the distribution of the airflow angle at blades of different C_F values along the blade height direction. As C_F decreases, the blade inlet airflow angle gradually increases, and at $C_F=0.43$, the angle increases considerably or even becomes positive. Figure 19 shows the relative velocity contour in front of the blade for different C_F values. It is seen that the change in the airflow angle changes the stagnation area in front of the blade. The sharp increase in the stagnation area at $C_F=0.5$ and 0.43 increases the pressure at the blade leading edge.

To compare the impact of rotational speed on the transient pressure field, the results at $0T$ for different flow coefficients are analyzed. Figure 20 displays the distribution of ΔC_p at

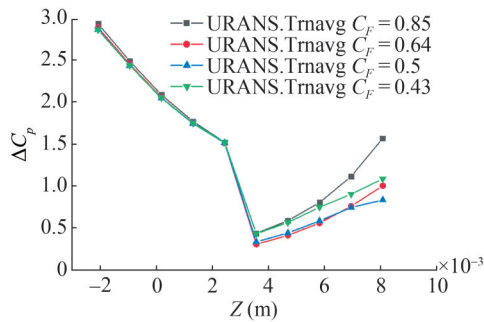


Figure 17 ΔC_p axial distribution of time-averaged URANS from the vane trailing edge to the blade leading edge at different C_F values and a fixed mass flow

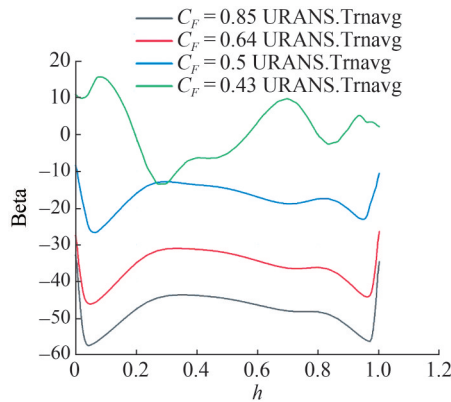


Figure 18 Distribution of the airflow angle at the blade along the blade height direction for different C_F values

0T for various flow coefficients. The axial distribution patterns of transient ΔC_p and time-averaged ΔC_p are closely aligned, except in the rim seal region. At $C_F=0.43$, the larger inlet airflow angle amplifies the influence of the blade potential flow pressure field. However, the impact on the vane side pressure remains minimal. Figure 21 shows the circumferential pressure distribution of the vane trailing

edge and the blade leading edge at different C_F values at 0T. It is seen that the speed has less influence on the pressure on the side of the vane compared to the side of the blade.

Figure 22 displays the axial distribution of ΔC_p at different times within one blade passing period for different C_F values, using time-averaged URANS as a reference. When $C_F=0.85$ and 0.64, the pressure distribution on the vane side is almost constant over time; this result indicates that the blade potential flow pressure field has minimal influence on the upstream vane. However, on the blade side, distinct variations in pressure are observed over time due to the effects of rotor–stator interaction. For the case of $C_F=0.43$, where the airflow angle at the blade inlet is larger, the mainstream and the wake are stalled on the larger surface area of the blade suction side. Thus, the pressure values on the rotor side vary slightly over time. However, because of the different relative circumferential positions of the rotor and vane, the blade pressure field has varying effects on the vane side.

3.3 Influence of C_F (changing the mass flow) on mainstream pressure

The pressure field is analyzed for the different flow coefficients caused by different mass flows with a fixed rotational speed. Figure 23 shows the time-averaged URANS results of the axial distribution of ΔC_p at different C_F values. For the vane side, ΔC_p decreases with decreasing mass flow because the high-pressure area on this side originates from the velocity loss of vane trailing, and when the mainstream flow rate is larger, the total pressure is larger, and the static pressure value formed after the velocity loss is larger. On the blade side, ΔC_p is not only determined by the mainstream flow rate but also influenced by the airflow angle at the blade inlet. For $C_F=0.85$, 0.64, and 0.5, ΔC_p decreases on the blade side as the mainstream flow rate decreases. However, when C_F is 0.3, ΔC_p suddenly increases on the

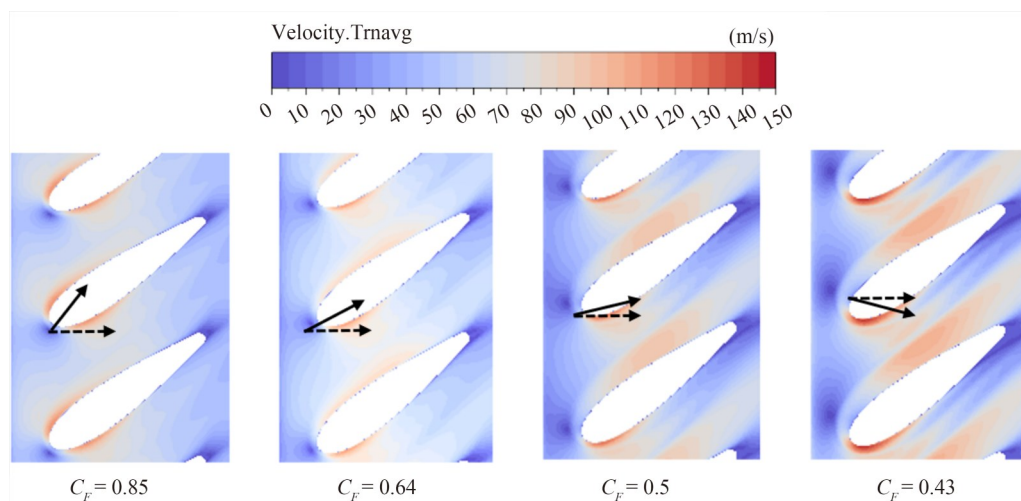


Figure 19 Relative velocity contour at the blade leading edge for different C_F values of time-averaged URANS results

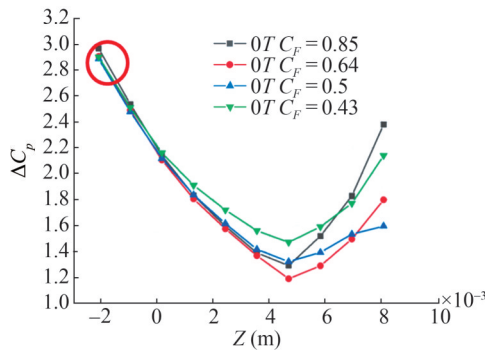


Figure 20 Axial distribution of ΔC_p at time $0T$ for different C_F values

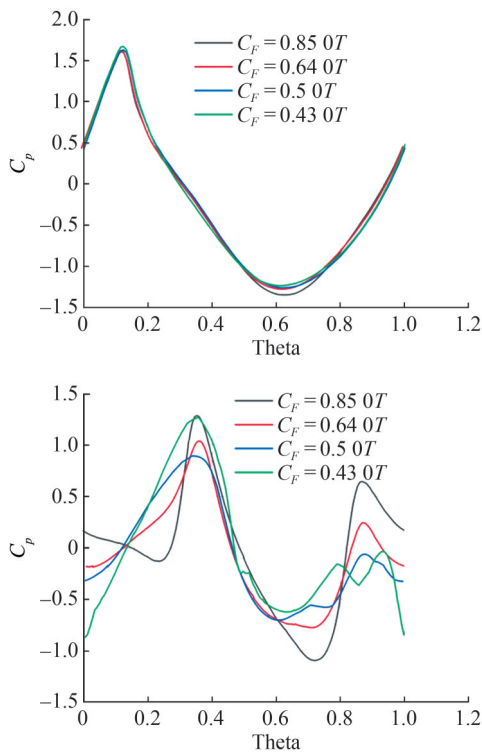


Figure 21 C_p circumferential distribution at the vane trailing edge and blade leading edge at different C_F values

blade side even higher than on the vane side. Referring to the analysis above of changing the rotational speed, the sharp increase in the airflow angle at the blade inlet can increase the pressure on the blade side.

Figure 24 displays the axial distribution of ΔC_p at different times within one blade passing period for different C_F values. At small airflow angles at the blade leading edge, corresponding to $C_F=0.85$, 0.64 , and 0.5 , the pressure distribution on the vane side is almost unchanged over time, indicating that the influence of the blade's potential flow pressure field on the upstream vane is small, and the pressure value on the vane side is larger than the pressure value on the blade. The influence of rotor–stator interaction on the

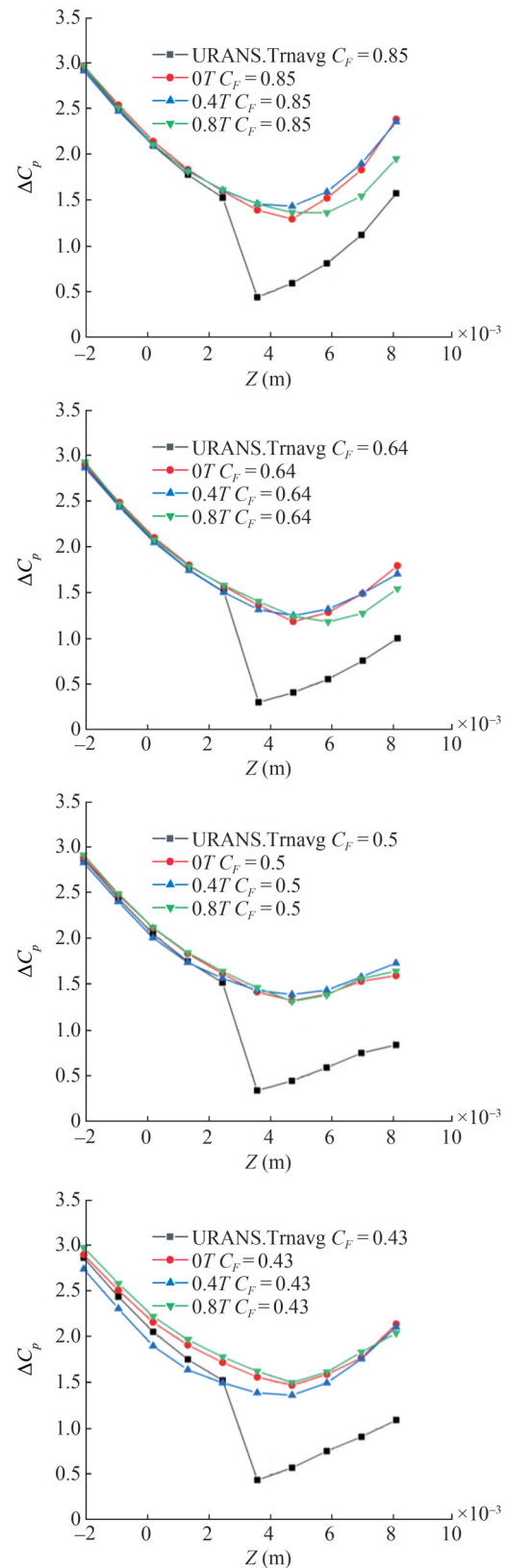


Figure 22 Axial distribution of ΔC_p at different C_F values

blade side is large, and the blade side pressure varies considerably over time. At large airflow angles at the blade

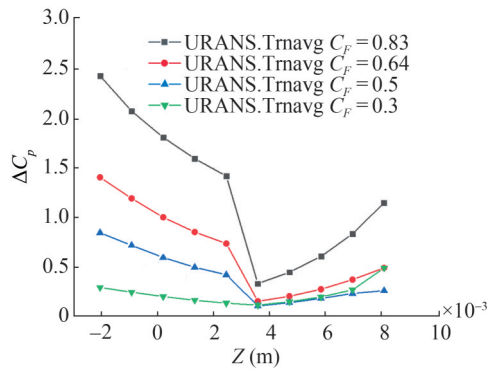


Figure 23 ΔC_p axial distribution of time-averaged URANS from the vane trailing edge to the blade leading edge at different C_F values for a fixed rotational speed

leading edge, corresponding to $C_F=0.3$, the increased incoming flow angle increases the stagnation area on the blade side. This result increases the pressure on the blade side because of a greater amount of mainstream flow stagnation. Consequently, the pressure is higher on the blade side than on the stator side. Because of the larger airflow angle, the pressure variation on the blade side is relatively small over time. In contrast, the pressure distribution on the vane side is considerably influenced by the rotor-induced potential flow pressure field and thus shows noticeable time variation.

3.4 Experimental results under variable operating conditions

Based on the analysis of the numerical simulation of the pressure field under different C_F conditions obtained by changing the rotating speed or mass flow, an experimental study of changing C_F in two ways is conducted. Because of the difficulty of measuring the pressure on the blade side, only the pressure on the vane side is measured. Figure 25 shows the circumferential distribution of the dimensionless pressure at the vane trailing edge for different C_F values obtained by varying the mass flow at a fixed speed of 4 000 r/min. The high pressure and circumferential differential pressure at the vane trailing edge decrease gradually with decreasing mass flow, consistent with the trend of the numerical simulation. Figure 26 shows the experimental results of the dimensionless pressure distribution at the vane trailing edge for different rotating speeds at two main flow rates. For two fixed main flow rates, different rotating speeds have no effect on the pressure distribution at the vane trailing edge and essentially no effect on the circumferential pressure difference at this edge. When the mass flow is fixed, the vane trailing pressure distribution is unaffected by the rotating speed, consistent with the regularity of the numerical simulation results.

Although changing the mass flow and rotating speed can change C_F , these two approaches affect the annulus

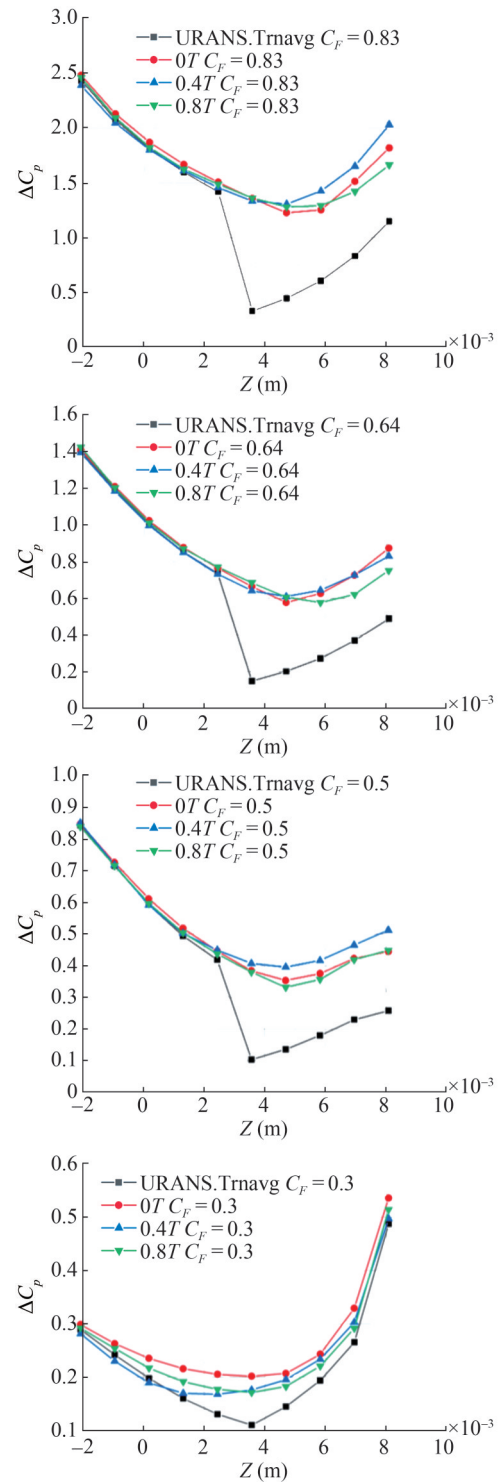


Figure 24 Axial distribution of ΔC_p at different C_F values

pressure distribution differently. Experimentally measuring the pressure distribution on the vane trailing edge reveals that this pressure distribution is mainly determined by the mass flow rather than rotating speed. The regularity of the numerical simulation is experimentally derived and confirmed.

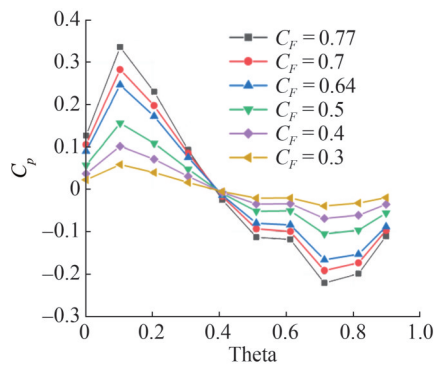


Figure 25 Circumferential distribution of C_p at the vane trailing edge under different mass flow rates at a fixed speed of 4 000 r/min

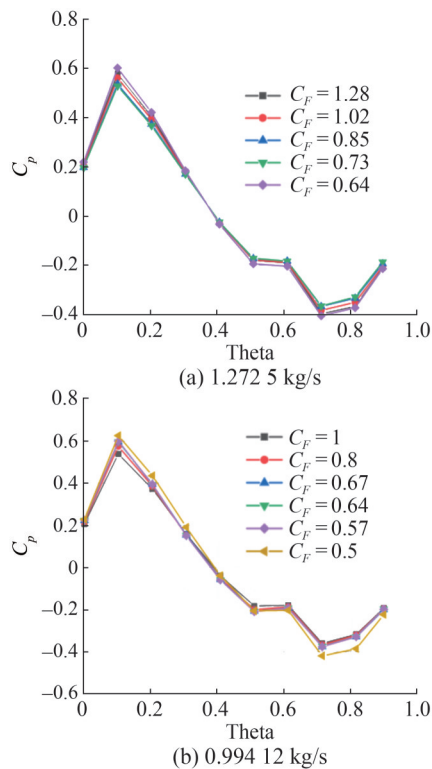


Figure 26 Circumferential distribution of C_p at the vane trailing edge under different rotating speeds at fixed mass flow rates

4 Conclusions

This paper analyzes the pressure field of the turbine interstage under the influence of rotor–stator interaction using numerical simulations and experiments for different C_F conditions obtained by changing the mass flow rate or the rotational speed. The following main conclusions are drawn.

1) Among the numerical simulation methods, the URANS method is the most suitable. The pressure at the blade's leading edge is influenced by the trailing and mainstream stagnation degrees at different times. The mainstream stagnation has a greater impact than the trailing stagnation. In the determined working condition, the changes in pressure

on the blade side are attributed to variations in the attack angle over time.

2) An analysis of the mainstream time-averaged pressure field under different flow coefficients shows that the mainstream pressure is primarily influenced by the mainstream flow rate and the angle of attack on the blade inlet. The vane side pressure decreases as the mainstream flow rate decreases. The blade side pressure is related to the mainstream flow rate and the angle of attack on the blade inlet. When the incidence angle is small, the blade side pressure decreases monotonically with decreasing mainstream flow rate. However, when the incidence angle is large, the blade side pressure sharply increases even at a low mainstream flow rate.

3) The analysis of the instantaneous pressure field shows that different flow coefficients obtained from the two methods lead to varying angles of attack on the blade inlet. For small incidence angles, the rotor–stator interaction causes substantial variations in the blade side pressure distribution over time while the vane side pressure remains relatively stable. For large incidence angles, the larger stagnation area reduces the temporal variations in blade side pressure and enhances the pressure potential field, varying the vane side pressure distribution.

4) The effects of rotational speed and mainstream flow rate on pressure are different and need to be clearly defined before studying the effect of flow coefficients on pressure. Since the differential pressure is sometimes higher on the blade side than the vane side, judging the degree of ingestion only by the vane side pressure is imperfect.

Acknowledgments The authors wish to acknowledge the financial support of the National Natural Science Foundation Outstanding Youth Foundation (Grant No. 52122603), the National Science and Technology Major Project (J2019-III-0003–0046), and the cloud computing supported by the Beijing Super Cloud Computing Center.

Competing interest The authors have no competing interests to declare that are relevant to the content of this article.

References

- Beard PF, Gao F, Chana KS, Chew J (2017) Unsteady flow phenomena in turbine rim seals. *Journal of Engineering for Gas Turbines and Power* 139(3): 032501. DOI: 10.1115/1.4034452
- Bohn DE, Decker A, Ma H, Wolff M (2003) Influence of sealing air mass flow on the velocity distribution in and inside the rim seal of the upstream cavity of a 1.5-stage turbine. In *Turbo Expo: Power for Land Sea and Air* 36886: 1033–1040. <https://doi.org/10.1115/GT2003-38459>
- Bru Revert A, Beard PF, Chew JW, Bottenheim S (2021) Performance of a turbine rim seal subject to rotationally-driven and pressure-driven ingestion. *Journal of Engineering for Gas Turbines and Power* 143(8): 081025. <https://doi.org/10.1115/>

- 1.4049858
- Duda D, Jelínek T, Milčák P, Němec M, Uruba V, Yanovych V, Žitek P (2021) Experimental investigation of the unsteady stator/rotor wake characteristics downstream of an axial air turbine. *International Journal of Turbomachinery, Propulsion and Power* 6(3): 22. <https://doi.org/10.3390/ijtp6030022>
- Eymann S, Reinmüller U, Niehuis R, Förster W, Beversdorff M, Gier J (2002) Improving 3D flow characteristics in a multistage lp turbine by means of endwall contouring and airfoil design modification: Part 1—design and experimental investigation. In *Turbo Expo: Power for Land Sea and Air* 3610: 249–260. <https://doi.org/10.1115/GT2002-30352>
- Förster F, Sims-Williams D, Ingram G (2012) Reconstruction of the unsteady pressure field in a low speed linear cascade. In *Turbo Expo: Power for Land Sea and Air* 44748: 2653–2662. <https://doi.org/10.1115/GT2012-69156>
- Gao F, Chew JW, Marxen O (2020) Inertial waves in turbine rim seal flows. *Physical Review Fluids* 5(2): 024802. <https://doi.org/10.1103/PhysRevFluids.5.024802>
- Gaetani P (2018) Stator-rotor interaction in axial turbine: Flow physics and design perspective. In *Aircraft Technology*, 105–128. IntechOpen. DOI: 10.5772/intechopen.76009
- Gier J, Ardey S, Eymann S, Reinmüller U, Niehuis R (2002) Improving 3d flow characteristics in a multistage lp turbine by means of endwall contouring and airfoil design modification: Part 2—numerical simulation and analysis. In *Turbo Expo: Power for Land Sea and Air* 3610: 261–271. <https://doi.org/10.1115/GT2002-30353>
- Green T, Turner AB (1992) Ingestion into the upstream wheel-space of an axial turbine stage. In *Turbo Expo: Power for Land Sea and Air* 78934: V001T01A110. <https://doi.org/10.1115/1.2928368>
- Hills NJ, Chew JW, Turner AB (2002) Computational and mathematical modeling of turbine rim seal ingestion. *J. Turbomach.* 124(2): 306–315. <https://doi.org/10.1115/1.1456461>
- Hu J, Yang J, He X, Zeng W, Zhao Z, Yang J (2023) Transition of amplitude–frequency characteristic in rotor–stator interaction of a pump-turbine with splitter blades. *Renewable Energy* 205: 663–677. <https://doi.org/10.1016/j.renene.2023.02.008>
- Mahallati A, Sjolander SA (2013) Aerodynamics of a low-pressure turbine airfoil at low Reynolds numbers—Part II: Blade-wake interaction. *Journal of Turbomachinery* 135(1): 011011. <https://doi.org/10.1115/1.4006320>
- Matsunuma T (2006) Unsteady flow field of an axial-flow turbine rotor at a low Reynolds number. *Journal of Turbomachinery* 129(2): 360–371. <https://doi.org/10.1115/1.2464143>
- MacIsaac GD, Sjolander SA, Praisner TJ (2010) Measurements of losses and Reynolds stresses in the secondary flow downstream of a low-speed linear turbine cascade. In *Turbo Expo: Power for Land Sea and Air* 44021: 1299–1313. <https://doi.org/10.1115/1.4003839>
- Miller RJ, Moss RW, Ainsworth RW, Harvey NW (2003) The development of turbine exit flow in a swan-necked inter-stage diffuser. In *Turbo Expo: Power for Land Sea and Air* 36894: 863–875. <https://doi.org/10.1115/GT2003-38174>
- Owen JM (2011a) Prediction of ingestion through turbine rim seals—Part I: Rotationally induced ingress. *Journal of Turbomachinery* 133(3): 1–9. DOI: 10.1115/1.4001177
- Owen JM (2011b) Prediction of ingestion through turbine rim seals—Part II: Externally induced and combined ingress. *Journal of Turbomachinery* 133: 031006–1. DOI: 10.1115/1.4001178
- Palermo DM, Gao F, Chew JW, Beard PF (2019) Effect of annulus flow conditions on turbine rim seal ingestion. In *Turbo Expo: Power for Land Sea and Air* 58653: V05BT15A004. <https://doi.org/10.1115/GT2019-90489>
- Roy RP, Xu G, Feng J, Kang S (2001) Pressure field and main-stream gas ingestion in a rotor-stator disk cavity. In *Turbo Expo: Power for Land Sea and Air* 78521: V003T01A089. <https://doi.org/10.1115/2001-GT-0564>
- Roy RP, Feng J, Narzary D, Paolillo RE (2005) Experiment on gas ingestion through axial-flow turbine rim seals. *J. Eng. Gas Turbines Power* 127(3): 573–582. <https://doi.org/10.1115/1.1850499>
- Rose MG, Jenny P, Gier J, Abhari RS (2013) Experimentally observed unsteady work at inlet to and exit from an axial flow turbine rotor. *Journal of Turbomachinery* 135(6): 061017. <https://doi.org/10.1115/1.4023460>
- Sangan CM, Pountney OJ, Zhou K, Owen JM, Wilson M, Lock GD (2013a) Experimental measurements of ingestion through turbine rim seals—Part II: Rotationally induced ingress. *Journal of Turbomachinery* 135(2): 021013. <https://doi.org/10.1115/1.4006586>
- Sangan CM, Pountney OJ, Zhou K, Wilson M, Michael Owen J, Lock GD (2013b) Experimental measurements of ingestion through turbine rim seals—Part I: Externally induced ingress. *Journal of Turbomachinery* 135(2): 021012. DOI: 10.1115/1.4006609
- Scobie JA, Sangan CM, Owen JM, Wilson M, Lock GD (2014) Experimental measurements of hot gas ingestion through turbine rim seals at off-design conditions. *Proceedings of the Institution of Mechanical Engineers Part A: Journal of Power and Energy* 228(5): 491–507. <https://doi.org/10.1177/0957650914527273>
- Touil K, Ghenaiet A (2021) Flow unsteadiness and rotor-stator interaction in a two-stage axial turbine. *Proceedings of the Institution of Mechanical Engineers Part A: Journal of Power and Energy* 235(6): 1370–1393. DOI: 10.1177/0957650920961625
- Wang T, Xuan Y, Han X (2023) The effects of stator-rotor interaction on unsteady characteristics of turbine tip leakage flow. *Aerospace Science and Technology* 141: 108544. <https://doi.org/10.1016/j.ast.2023.108544>
- Yang H, He Q, Huang X, Yang M, Bi H, Wang Z (2022) Experimental and numerical investigation of rotor–stator interaction in a large prototype pump–turbine in turbine mode. *Energies* 15(15): 5523. <https://doi.org/10.3390/en15155523>
- Zheng Y, Gao Q, Yang H (2023) Forced response analysis of an embedded compressor rotor induced by stator disturbances and rotor–stator interactions. *Aerospace* 10(5): 398. <https://doi.org/10.3390/aerospace10050398>